EDF R&D

Fluid Dynamics, Power Generation and Environment Department
Single Phase Thermal-Hydraulics Group

6, quai Watier
F-78401 Chatou Cedex

Tel: 33 1 30 87 75 40
Fax: 33 1 30 87 79 16

MARCH 2024

code_saturne documentation

code_saturne version 8.0 tutorial: three 2D disks

contact: saturne-support@edf.fr

TABLE OF CONTENTS

Part I

Introduction

1 Introduction

1.1 code_saturne short presentation

code_saturne is a system designed to solve the Navier-Stokes equations in the cases of 2D, 2D axisymmetric or 3D flows. Its main module is designed for the simulation of flows which may be steady or unsteady, laminar or turbulent, incompressible or potentially dilatable, isothermal or not. Scalars and turbulent fluctuations of scalars can be taken into account. The code includes specific modules, referred to as "specific physics", for the treatment of lagrangian particle tracking, semi-transparent radiative transfer, gas, pulverized coal and heavy fuel oil combustion, electricity effects (Joule effect and electric arcs) and compressible flows. code_saturne relies on a finite volume discretization and allows the use of various mesh types which may be hybrid (containing several kinds of elements) and may have structural non-conformities (hanging nodes).

1.2 About this document

The present document is a tutorial for code_saturne version 8.0. It presents test cases and guides the future code_saturne user step by step into the preparation and the computation of the cases. It focuses on code_saturne SYRTHES coupling feature allowing to run simulations taking into account conjugate heat transfers.

The test case directories, containing the necessary meshes and data are available in the `examples/4-2Ddisks` directory in code_saturne source directory.

This tutorial focuses on the procedure and the preparation of the code_saturne and SYRTHES computations with or without SALOME. For more elements on the structure of the code and the definition of the different variables, it is higly recommended to refer to the user manual.

The first part is this introduction. In the second part, the configuration, geometry, data settings, numerical parameters are described for a solid computation alone, a fluid computation alone and finally for a coupled fluid/solid computation. In the third part, a detailed description of the steps to be followed is given.

1.3 code_saturne copyright informations

Part II

Three 2D disks

1 Study description

1.1 Objective

This tutorial case focuses on code_saturne coupling feature with SYRTHES, an open source code solving thermal conduction and radiative transfer problems in solids, which is one of the way to model conjugate transfer phenomenon with code_saturne.

The aim is to simulate the natural convection flow of air inside a sheath containing three electric wires. A 2D cross section of the sheath is considered. Thermal conduction inside the three wires is solved with SYRTHES and coupled with the flow solved with code_saturne.

In a first step, uncoupled solid and fluid calculations will be set and run separately, and only in a second step, a coupled solid/fluid calculation. The data settings used for the uncoupled step can largely serve as basis for the coupled step.

1.2 Study creation and preparation

First, create the 🗀 `3_2D_DISKS` study directory, with two subdirectories 🗀 `FLUID` and 🗀 `SOLID`. Two ways are available:

- Create coupled study with SALOME module CFDSTUDY as described on Figure II.1.

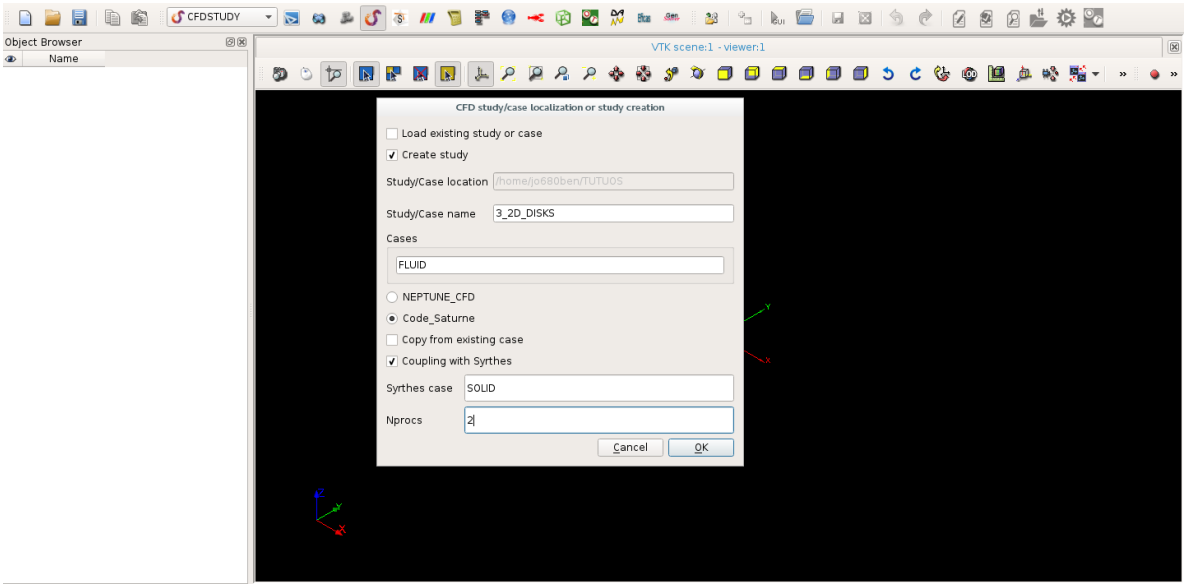

Figure II.1: Coupled study creation with SALOME module CFDSTUDY.

At this step, it is possible to choose the total number of processes that will be used for a coupled computation. Here 2 is chosen.

- Create coupled study in a terminal with the following command line:

```
$ code_saturne create -s 3_2D_DISKS -c FLUID --syrthes SOLID
```

Second, mesh files have to be copied in the study. The fluid mesh has to be copied in the subdirectory 🗀 `MESH`. The solid mesh must be copied in the subdirectory 🗀 `SOLID`.

Third, to set up the solid settings, launch SYRTHES Graphical User Interface (GUI) in SALOME by

opening SYRTHES module[1] or in a terminal by typing `$ syrthes.gui &` inside the subdirectory SOLID.

Fourth, to set up the fluid settings, launch the code_saturne GUI in SALOME or in a terminal inside the subdirectory FLUID/DATA (script SaturneGUI) for the fluid computation alone.

1.3 Geometry

As said above, the simplified configuration represents a 2D cross section of the electric wires inside the sheath. The 3 wires inside the sheath are represented as 3 disks inside a larger ring and it is assumed that the 3 disks are in contact with an air flow inside the electric sheath.

The geometry is shown on figure II.2.

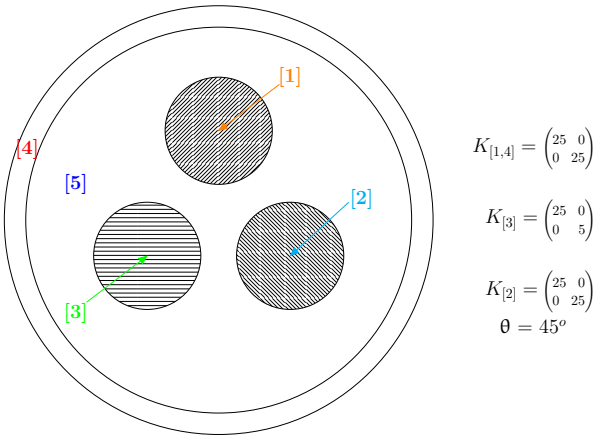

Figure II.2: Geometry with the solid domain (parts $1, 2, 3, 4$) and the fluid domain (part 5).

1.4 Meshes characteristics

- **Mesh of the solid domain:**

The mesh of the solid part contains 11688 nodes (P_1 discretization) and 5688 elements. Materials properties and boundary conditions are specified in the next part using the references given on figure II.3.

Type: unstructured mesh
Mesh generator used: SIMAIL

- **Mesh of the fluid domain:**

The fluid mesh contains 3866 nodes. **Mesh Quality Criteria** run type in the code_saturne Graphical User Interface (**Calculation management**, **Pepare batch calculation** section) can be used to check the quality of the mesh and to help to identify the references associated to the boundary conditions (1 is used for all solid boundaries, 2 for the front fluid face, and 3 for the back fluid face). See figure II.4.

Type: unstructured mesh
Mesh generator used: SIMAIL

[1] Once a .syd file is present, you can right-click on it in CFDSTUDY object browser and launch SYRTHES GUI from the contextual menu.

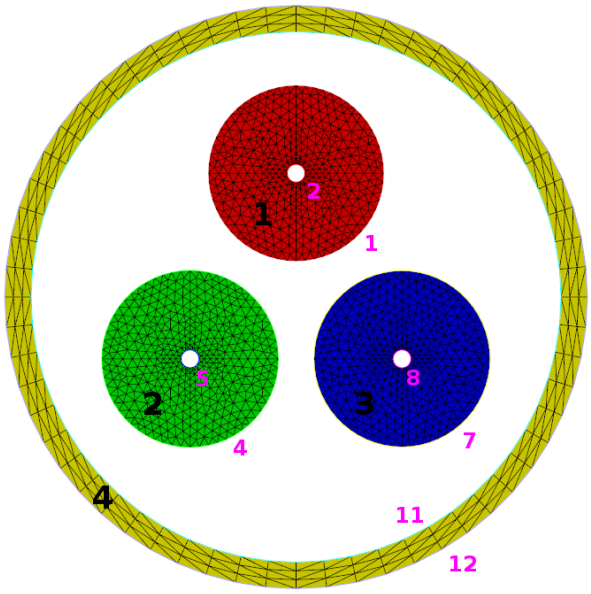

Figure II.3: References for the solid interior zones (black) and solid boundary zones (pink).

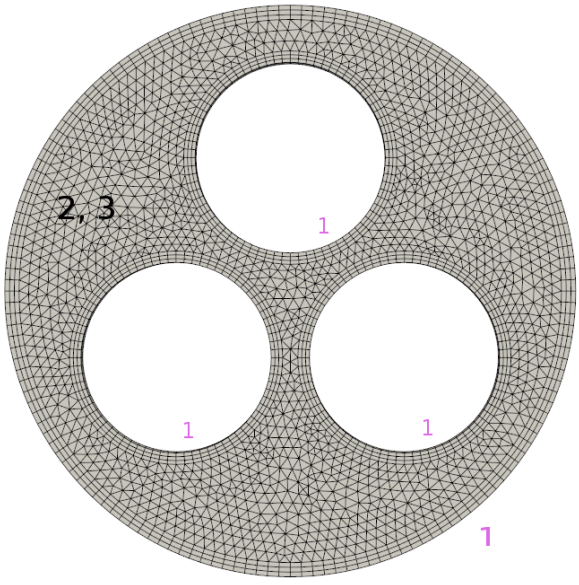

Figure II.4: fluid boundary faces references

1.5 Data settings for solid and fluid domains

- **Solid domain:**

Material properties for the solid domain are given by part (1 to 3 for the electric wires and 4 for the disk for the electric sheath) in the subsequent table:

	Conductivity type	**Values** (W/m/°C)	**Volume reference**
Disk 1	Isotropic	$k_{11} = 25$	1
Disk 2	Orthotropic	$k_{11} = 25$; $k_{22} = 5$	2
Disk 3	Anisotropic	$k_{11} = 25$; $k_{22} = 5$ $\alpha = 45^o$	3
Disk 4	Isotropic	$k_{11} = 25$	4

Physical properties	**Values**
Density $[\rho]$	7700 (kg/m^2)
Specific heat $[C_p]$	460 $(J/kg/m^3)$

Initial and boundary conditions for the solid domain should be defined as follows:

Initial conditions	
Temperature condition	$T_{ini,s} = 20$°C

Boundary conditions	**Value**	**Surface reference**
Heat exchange conditions $(q_{w,ext})$	$T_{ext} = 90$°C. ; $h_{ext} = 1000(W/m^2.K)$	2 or 5 or 8

For the coupled case, nothing needs to be changed in this part.

- **Fluid domain:**

Some characteristics of the air flow that should be checked in the GUI are given hereafter:

Modeling feature	**choice**
Time step	constant in time and uniform in space
Turbulence model	$k - \varepsilon$ Linear Production
Thermal model	Temperature (°C)

In the air flow, density is a function of the temperature and gravity force is taken into account. The 3 disks, which are warmer than the air flow, generate a temperature difference creating a fluid movement. The warmer air flow is moving to the top and the colder air flow to the bottom of the fluid domain.

The density follows an ideal gas law that can be specified in the GUI:

$$\rho = \frac{p_0}{R_g \ (T + 273.15)} \tag{II.1}$$

where ρ is the density, T is the temperature (°C), $R_g = 287$ $(m^2.s^{-2}.K^{-1})$ the ideal gas constant and $p_0 = 101325$ (Pa) the reference pressure (atmospheric pressure).

Initial conditions are defined below:

Initial conditions	
Temperature condition	$T_{ini,f} = 20$°C.

Symmetry conditions are imposed on the front and back face, and walls are set on all remaining faces (those coupling fluid and solid domain).

Boundary conditions	Values	Surface reference
Walls (Heat exchange $q_{w,ext}$)	T_{ext}(°C) ; $h_{ext}(W/m^2.K)$	1
Symmetry		2 or 3

For the fluid computation alone $T_{ext} = 30$°C and $h_{ext} = 10W/m^2.K$ while for the coupled case, simply set $T_{ext} = 0$°C and $h_{ext} = 0W/m^2.K$.

2 Computations of the 3 2D disks configuration

If not already done, set all data necessary to run the fluid and solid part configurations separately based on the description of the previous section. All the parameters necessary to this study can be defined through code_saturne GUI and SYRTHES GUI respectively.

2.1 Time stepping parameters

Set now all the time stepping parameters on both sides: code_saturne and SYRTHES.

Time stepping parameters of solid computation	
Reference time step	10 (s)
Number of iterations	100

Time stepping parameters of fluid computation	
Reference time step	0.1 (s)
Number of iterations	100

These time stepping parameters will be set to run the fluid and solid computations **independently from one another**.

2.2 Output management

Standard options for output management will be used. Only one monitoring point will be created for the solid conduction computation at the following coordinates:

Probe	x (m)	y (m)
1	0.003	-1.2

The output frequency for this probe can be every 10 time steps.

The temperature field can be saved every 25 time steps.

2.3 Results for the uncoupled cases

The velocity and **temperature** field can be post-processed with ParaVis.

As examples, Figure II.5 shows the evolution of the temperature in the solid domain and Figure II.6 the evolution of the temperature in the fluid domain without **Conjugate heat transfer**.

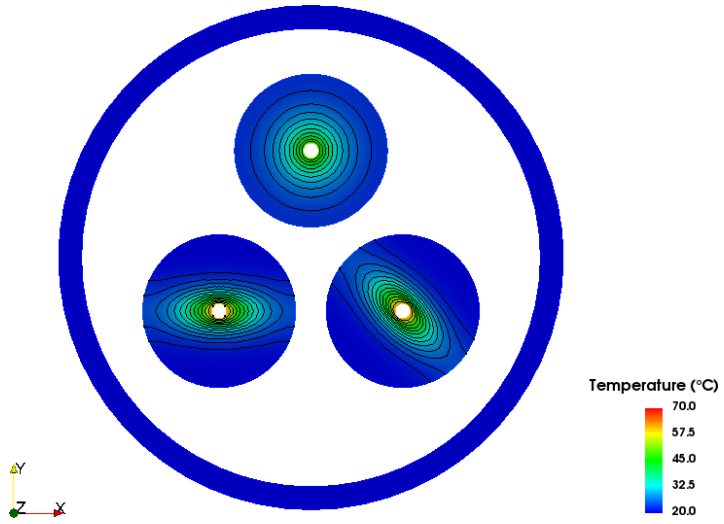

Figure II.5: The temperature evolution in the **solid domain without coupling method**

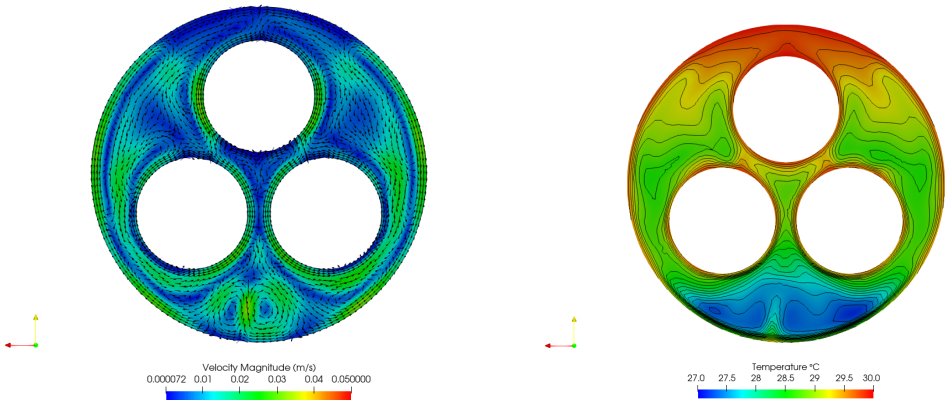

Figure II.6: The temperature evolution in the **fluid domain without coupling method**

2.4 Parameters for the coupled computation

Conjugate heat transfer has to be enabled and the coupling interfaces specified on both sides.

Then time stepping parameters have to be modified to be able to see the effect of the conjugate heat transfer phenomenon between the solid and fluid domains. For this reason, we increase the number of iterations and the reference time step for the fluid and the solid part[2].

Time stepping parameters of solid computation	
Reference time step	0.5 (s)
Number of iterations	600

Time stepping parameters of fluid computation	
Reference time step	0.5 (s)
Number of iterations	600

[2]By default, the smallest number of iterations will be used to drive the coupling computation. If we choose a number of iterations of 10000 for the fluid domain and 5000 for the solid domain, the coupling computation will be stopped after 5000 instead of 10000.

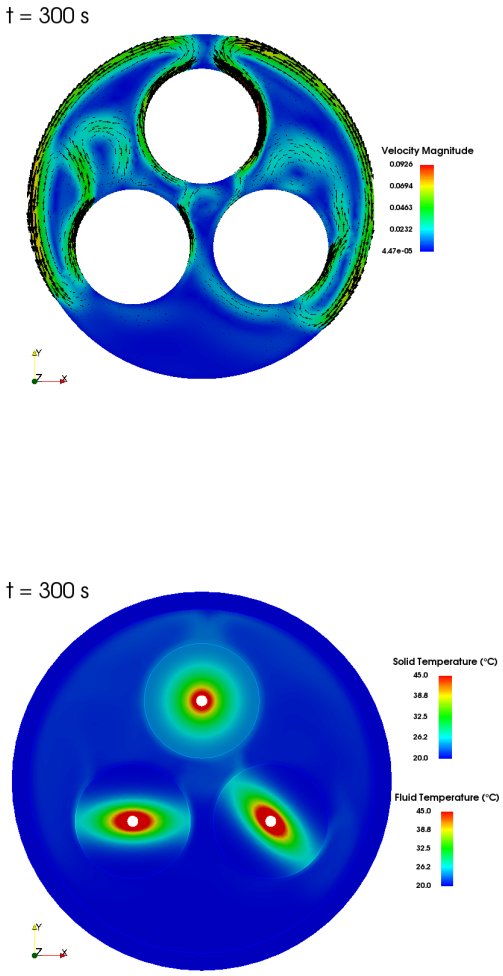

Figure II.7: Evolution of temperature and velocity magnitude

The **Improved pressure interpolation in stratified flow** option has to be checked in code_saturne GUI (**Numerical parameters**, **Global parameters** section).

2.5 Results for the coupled computation

As an example, Figure II.7 shows the evolution of the temperature in the solid and fluid area with the **conjugate heat transfer** enabled. The natural convection in the fluid domain due to the temperature difference imposed by the solid disks is clearly visible with the velocity displayed as a vector field.

Part III

Step by step description

1 Detailed tutorial step by step

1.1 Creation of the study in a terminal

• **Step 1**: check the post-install required for coupling code_saturne with SYRTHES.
The first step is to check the post-install required for coupling with SYRTHES and verify if the SYRTHES •PATH is correctly known in the system environment. We just need to edit the batch file[1] name `code_saturne.cfg` as below:

```
$ vim <install-prefix>/etc/code_saturne.cfg
>### Set the location to the SYRTHES installation directory.
> syrthes = <install-prefix-syrthes>
```

• **Step 2**: source the `syrthes.profile` file in your user environment.
Before using SYRTHES alone, you have to copy and source this file to define SYRTHES environment variables (like `$SYRTHES4_HOME`) in your terminal, as follows:

```
$ cp <install-prefix-syrthes>/bin/syrthes.profile .
$ source syrthes.profile
$ echo $SYRTHES4_HOME (to check the SYRTHES PATH in your environment)
```

After having defined correctly your environment, to be able to launch a coupling computation code_saturne-SYRTHES or a SYRTHES computation alone, you just have to create the coupling study directory.

• **Step 3**: create the 🗀`3_2D_DISKS` study directory, and the two case subdirectories 🗀`FLUID` and 🗀`SOLID`.

This is done using the standard command:

```
$ code_saturne create -s 3_2D_DISKS -c FLUID --syrthes SOLID
> code_saturne 3.0 study/case generation
> o Creating study '3_2D_DISKS' ...
> o Creating case 'FLUID' ...
> SYRTHES4 home directory:  <install-prefix-syrthes>
> MPI home directory:  /usr
>
>************************************************
> SOLID : creating SYRTHES case ...
> <install-prefix-syrthes>
> OK !
>************************************************
```

• **Remark**: The fluid mesh must be copied in the directory 🗀`MESH`. The solid mesh must be copied in the subdirectory 🗀`SOLID`.

[1] see the installation guide, name `install.pdf`, in `<install-prefix>/share/doc/code_saturne/` directory.

1.2 Launching SYRTHES

Preparation of SYRTHES computation alone can be one in 6 steps:

- **Step 1**: Launch SYRTHES module in SALOME or SYRTHES GUI in a terminal,
- **Step 2**: Create a **New Data File**,
- **Step 3**: Check the name of the mesh and convert this one in .syr format,
- **Step 4**: Define the initial and boundary conditions for the conduction problem,
- **Step 5**: Define the physical properties of each disk {1, 2, 3 and 4},
- **Step 6**: Running the SYRTHES computation alone.

• **Step 1**: launch SYRTHES module in SALOME or SYRTHES GUI in a terminal.
SYRTHES GUI can be launched by the following command lines in the solid subdirectory:

```
$ cd 3_2D_DISKS/SOLID/
$ syrthes.gui &
```

• **Step 2**: choose `New Data File` inside the pop-up window.

Figure III.1: Opening pop-up window for SYRTHES GUI in SALOME

1.3 Home Tab

Once `New Data File` is selected the Home Tab SYRTHES GUI appears.

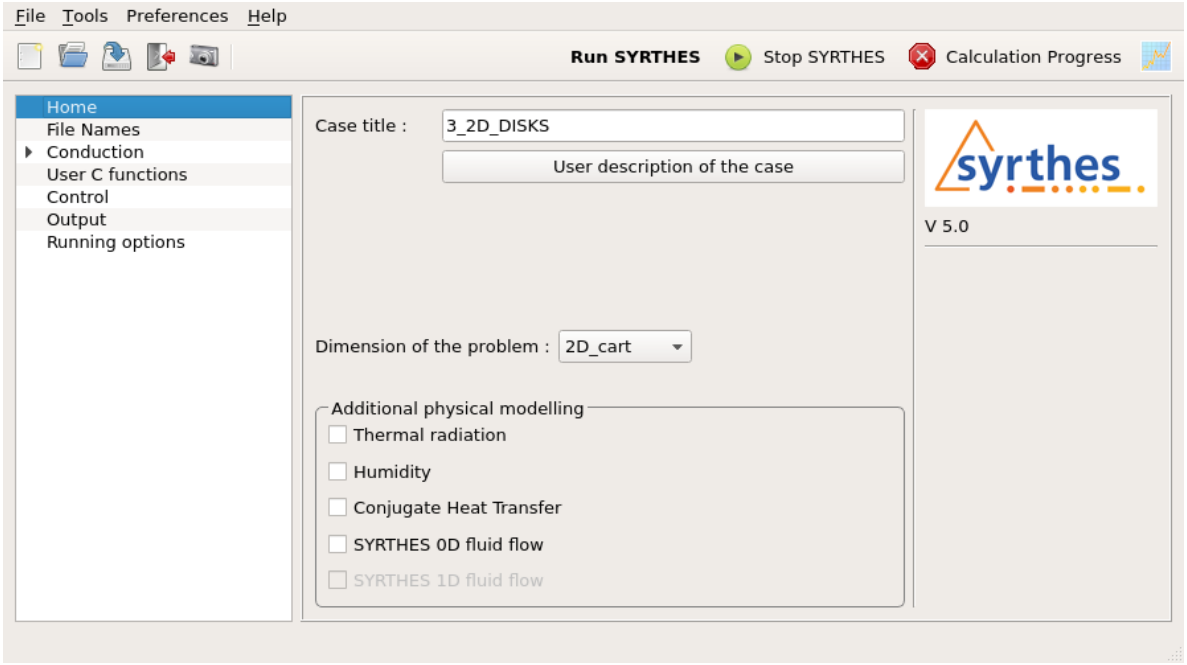

Figure III.2: Syrthes Home Tab

1.4 File Names Tab

• **Step 3**: Select the mesh
Add the solid mesh in **Conduction Mesh** field.Choose the 2D solid mesh file with the format `.des`.

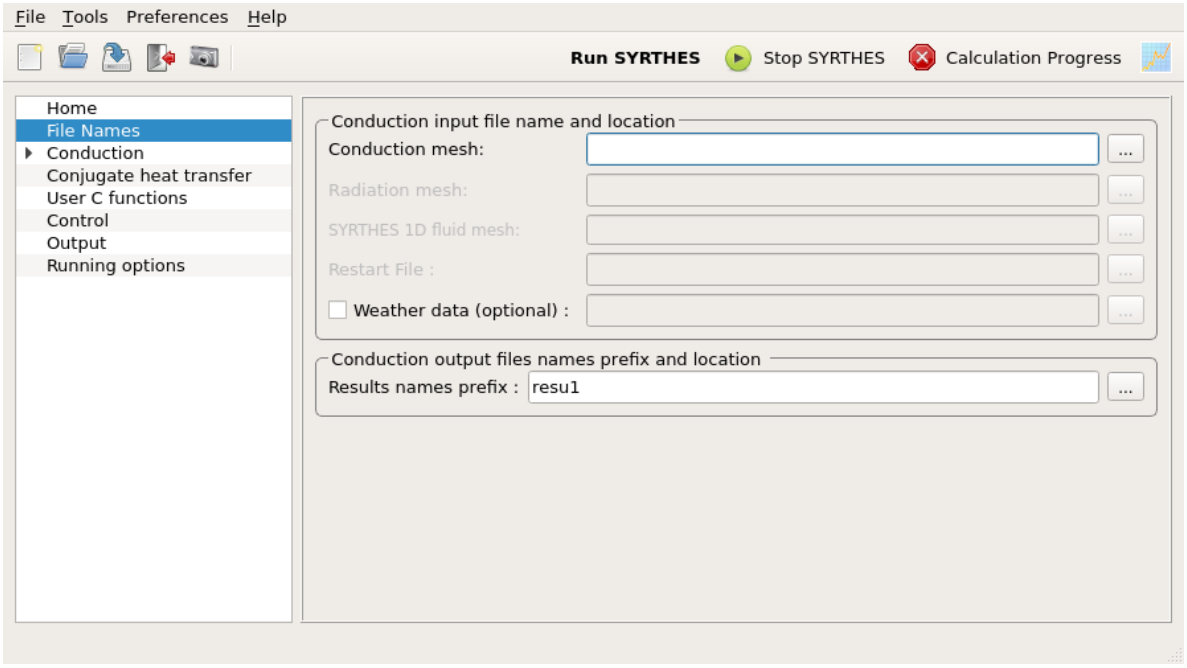

Figure III.3: File Names Tab - Conduction Mesh

The SYRTHES (GUI) directly converts the `.des` to the `.syr` format.

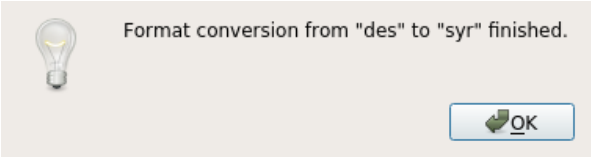

Figure III.4: Converting mesh format

• **Note:** Inside SYRTHES GUI, we can load the SIMAIL format `*.des` for the solid mesh. This one will be automatically transformed to the `*.syr` format.
It can also be done with the following command line:

```
$ convert2syrthes4 -m 3rond2d.des
```

• **Remark**: You can convert the `*.syr` format into a `*.med` format. Like that, you can load the `*.med` file inside SALOME, after having used this command line below:

```
$ syrthes4med30 -m 3rond2d.syr -o 3rond2d.med
```

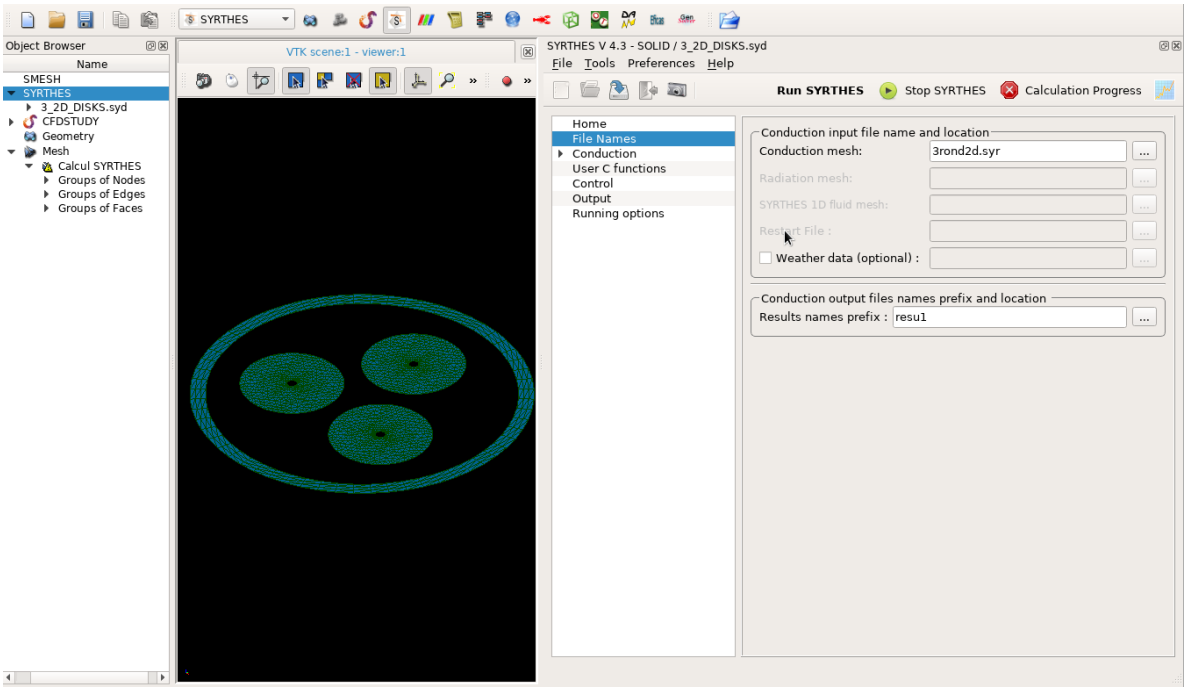

Figure III.5: Choose a name for the results files `.res`, `.his` and `.rdt`

1.5 Conduction Tab

• **Step 4**: Define initial and boundary conditions

Initial conditions Define the initial temperature conditions inside the different disks. The reference value '-1' means 'applied for the whole domain'.

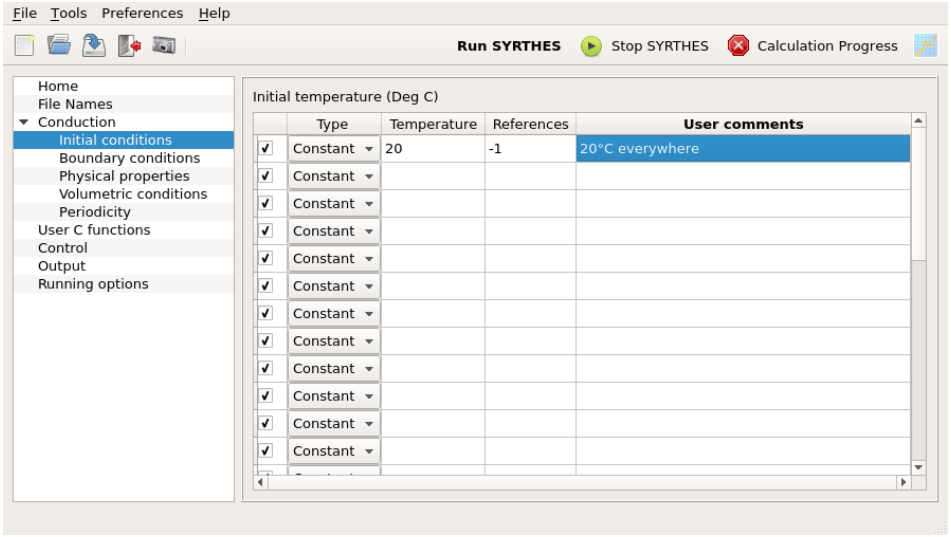

Figure III.6: Initial temperature conditions

Boundary conditions Define the temperature boundary conditions for the extern faces of the three disks as follows.

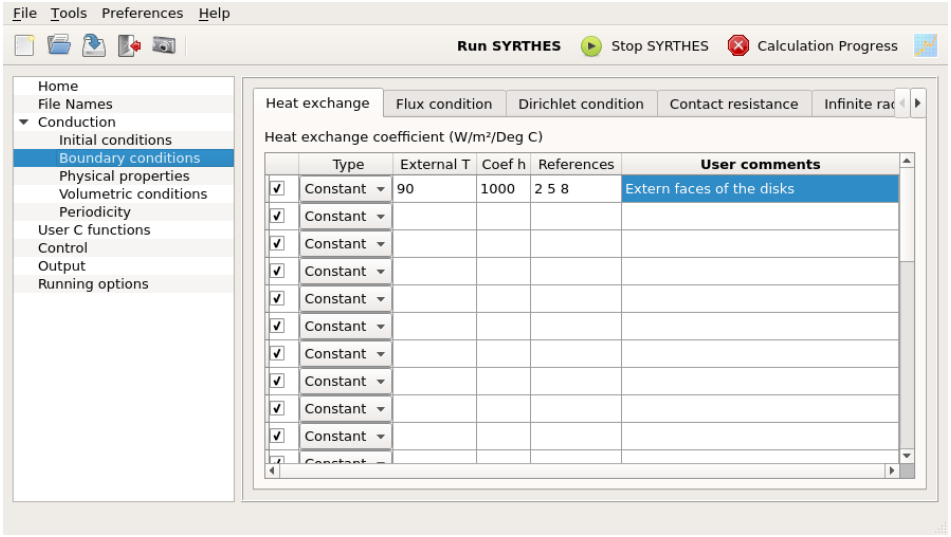

Figure III.7: Boundary conditions Heat exchange

• **Step 5**: Define physical properties for each disk

Physical properties Define Isotropic, Orthotropic and Anisotropic properties as follows.

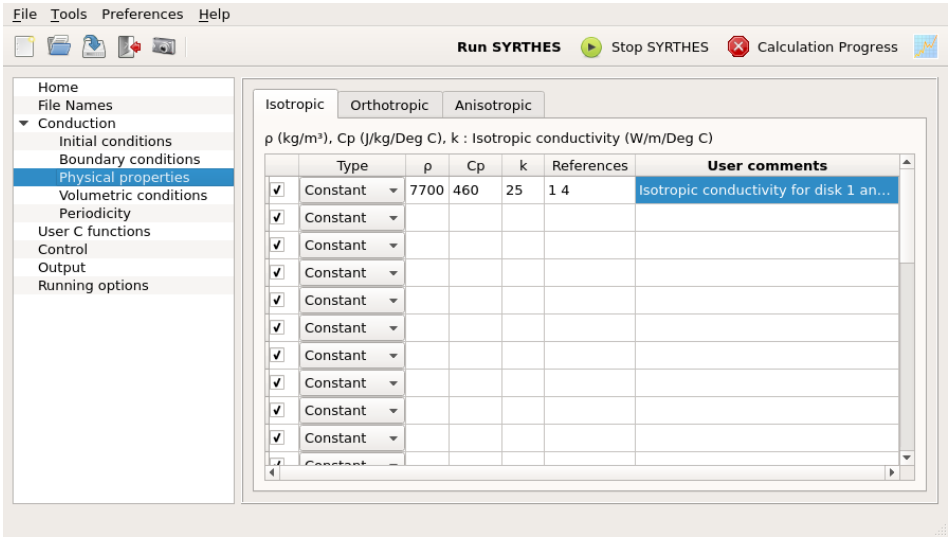

Figure III.8: Isotropic properties on disks 1 and 4

Define the physical properties for the disk 2 with orthotropic conductivity.

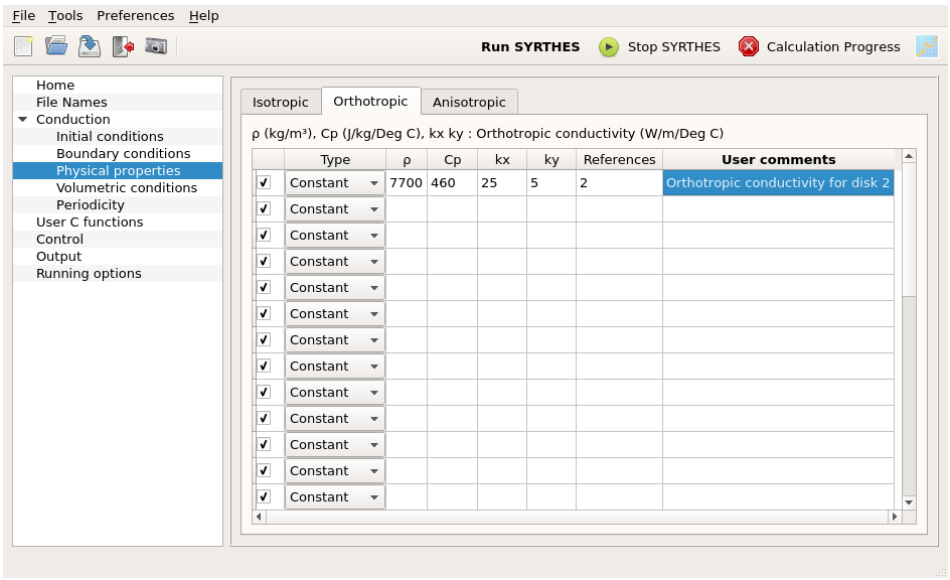

Figure III.9: Orthotropic properties on disk 2

• **Remark**: To correctly identify the volume references associated to a specific physical property, we can check the mesh regions directly inside ParaVis.
Define the Physical properties for the disk 3 with anisotropic conductivity.

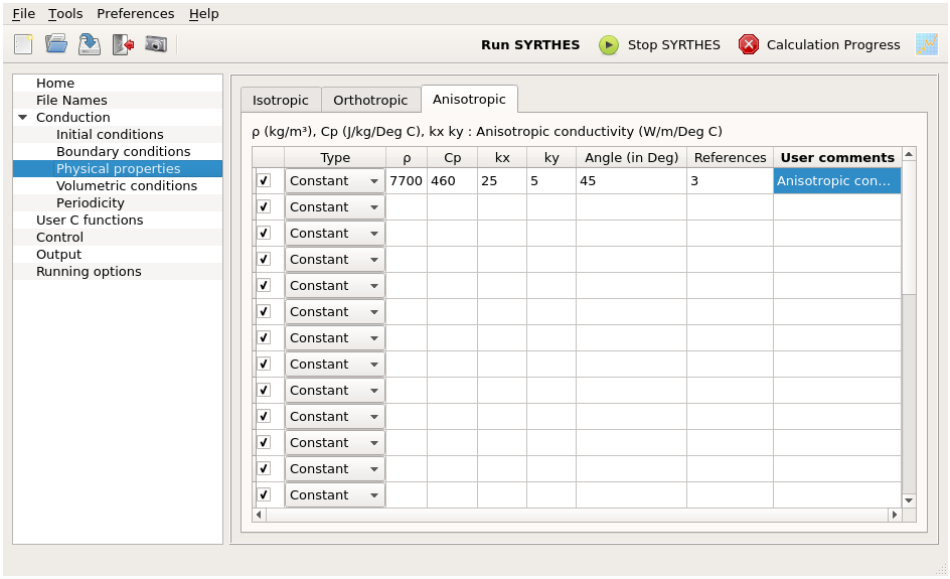

Figure III.10: Anisotropic properties on disk 3
Mesh

1.6 Control Tab

Define the global number of time steps and the time step for the 2D solid conduction computation.

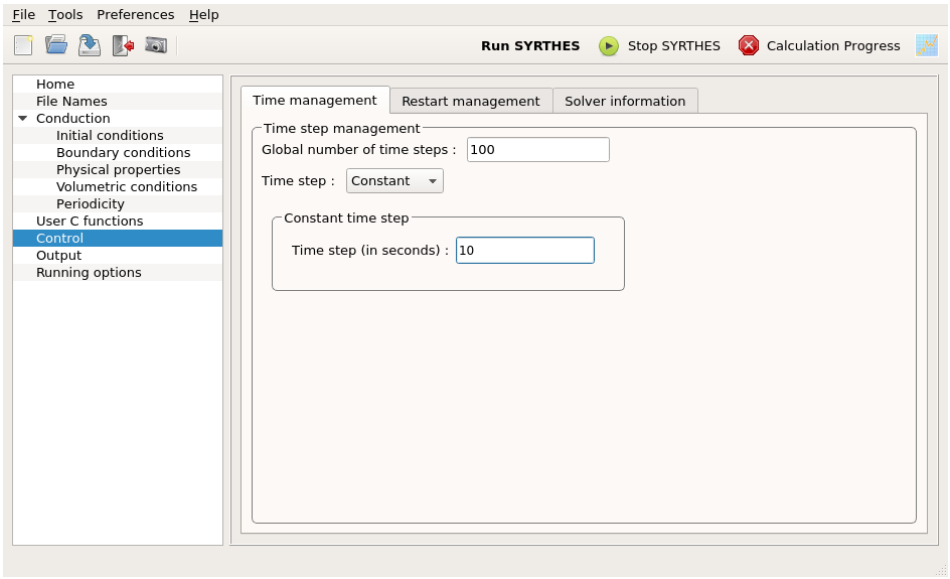

Figure III.11: Time management settings

1.7 Output

Define the probe coordinates for output management.

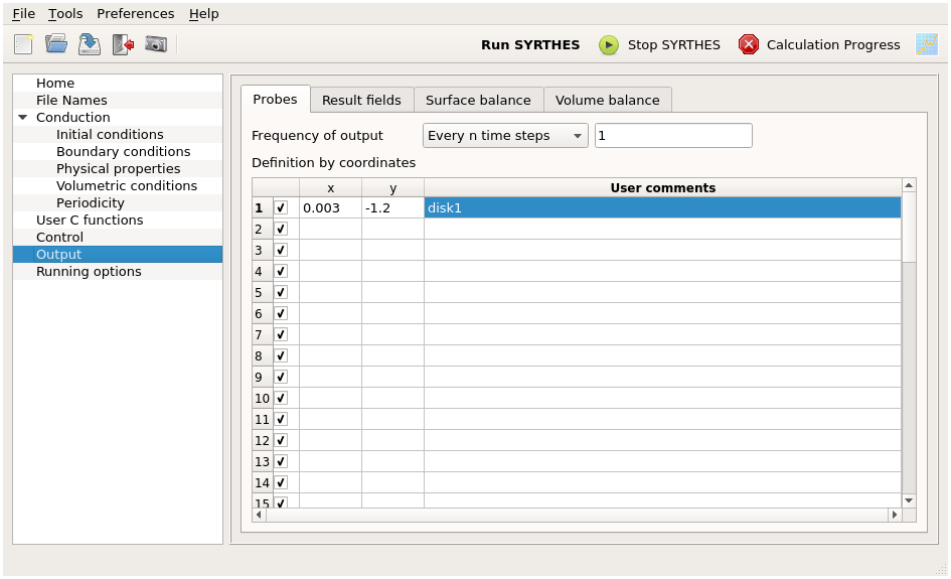

Figure III.12: Probes creation

Define the frequency at which the results fields are written

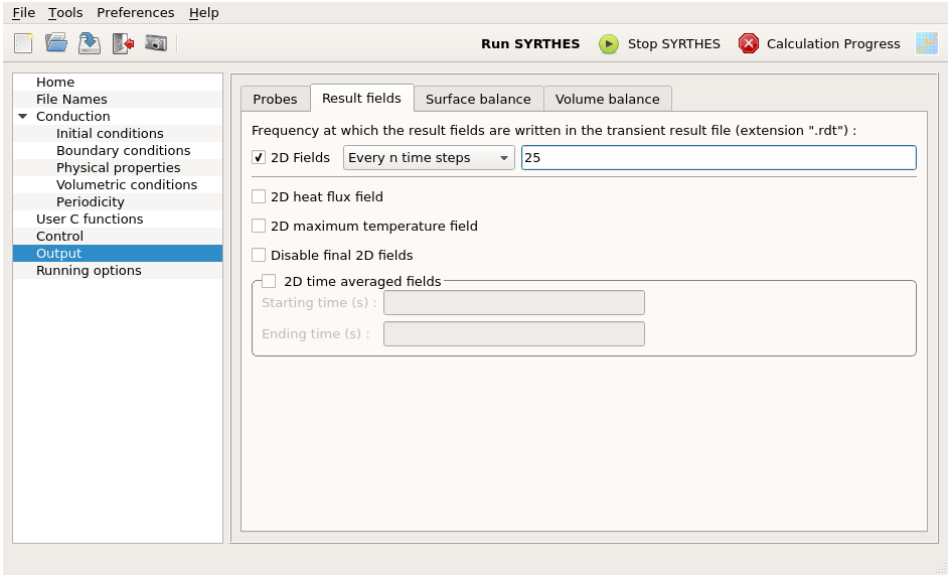

Figure III.13: Results fields window

1.8 Running options

• **Step 6**: Running SYRTHES computation alone.
Define the file name of the SYRTHES listing and the number of processors used.

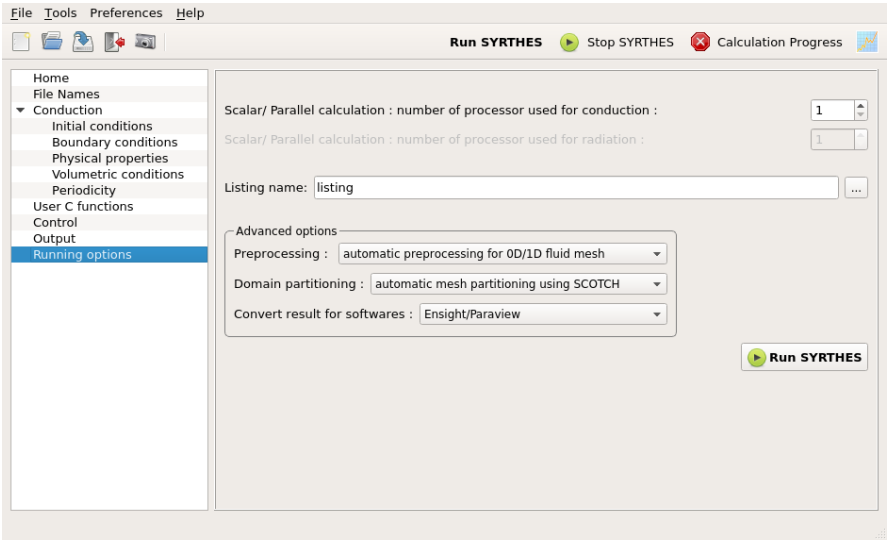

Figure III.14: Running options window

The following graph represents monitoring point the temperature value versus time during computation
.

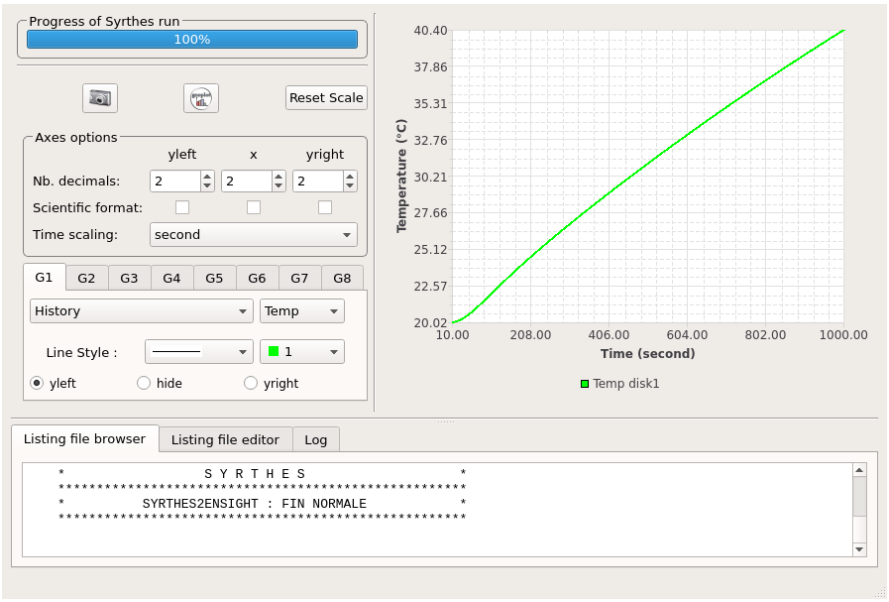

Figure III.15: Screenshot of the computation progress window

1.9 Preparing and launching the code_saturne computation alone

The main steps of the preparation of the fluid computation alone can be the following ones:
• **Step 1**: Launch code_saturne GUI from the object browser or the tool bar in SALOME module CFDSTUDY (or `./SaturneGUI` in command line),
• **Step 2**: Create a **New case**,
• **Step 3**: Check mesh quality by running a **Mesh quality criteria** calculation,
• **Step 4**: Define Mesh settings and Boundary zones
• **Step 5**: Define Calculation features
• **Step 6**: Define Volume conditions
• **Step 7**: Define Boundary conditions
• **Step 8**: Define the Time Settings
• **Step 9**: Define postprocessing
• **Step 10**: Run the code_saturne computation alone.

• **Step 1 & 2** : Create code_saturne `New case` and launch code_saturne

```
$ code_saturne create -s 3_2D_DISKS -c FLUID --syrthes SOLID
$ code_saturne gui &
```

1.10 Mesh Tab

• **Step 3**: Check mesh quality by running a **Mesh quality criteria** calculation. Choose the fluid mesh with code_saturne (GUI) and select the execution mode *Mesh quality criteria only*. Click on run computation to check mesh quality.

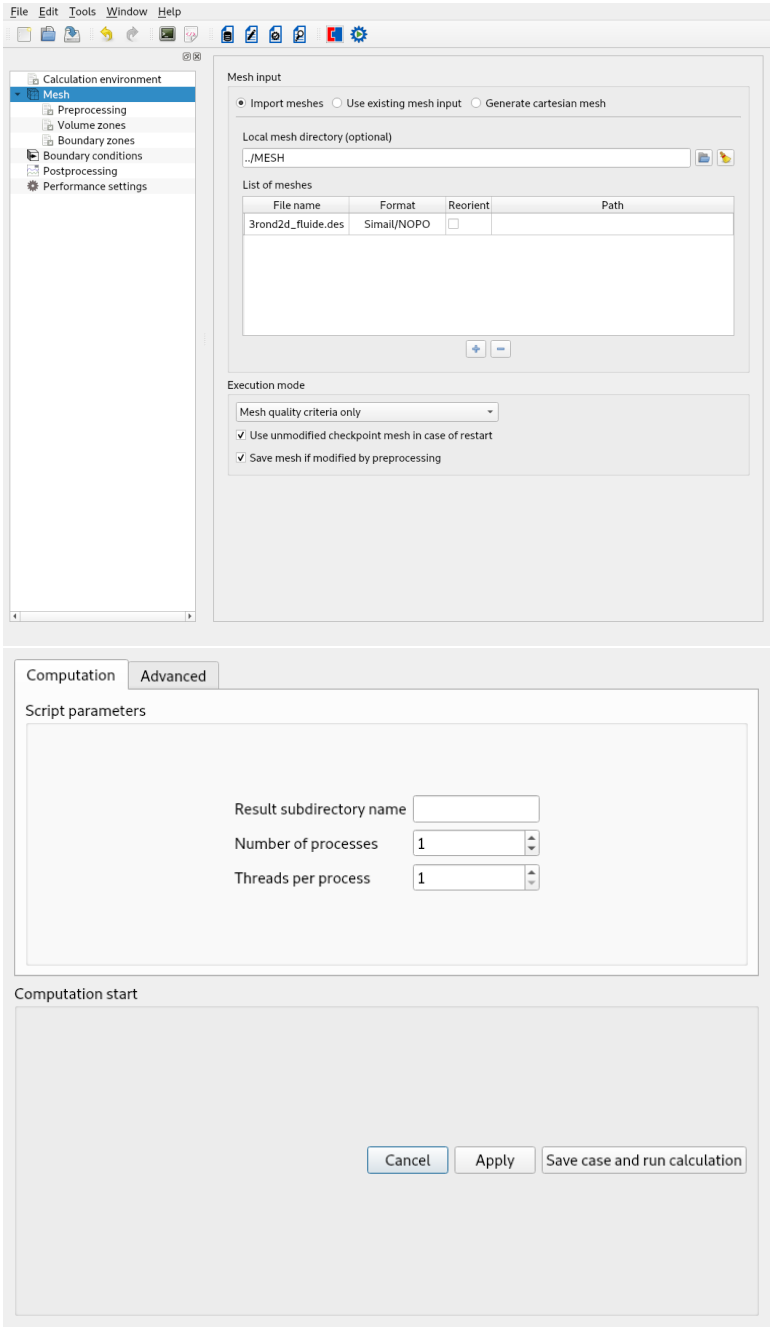

Figure III.16: Mesh quality criteria only computation

The generated *preprocessor.log* file, located in RESU will be used in the next step.

• **Step 4**: Define Mesh settings and Boundary zones
Select the option *Standard computation* as execution mode and set up physical properties as follows. Load the `preprocessor.log` file inside the code_saturne (GUI) to define boundary regions.

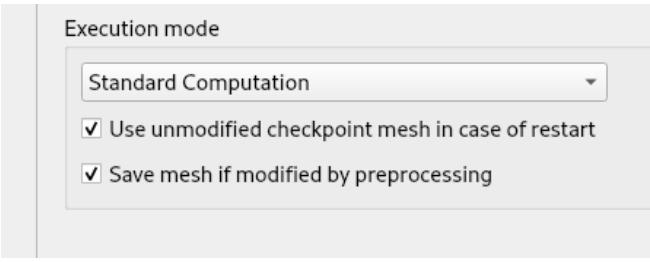

Figure III.17: Standard computation execution option

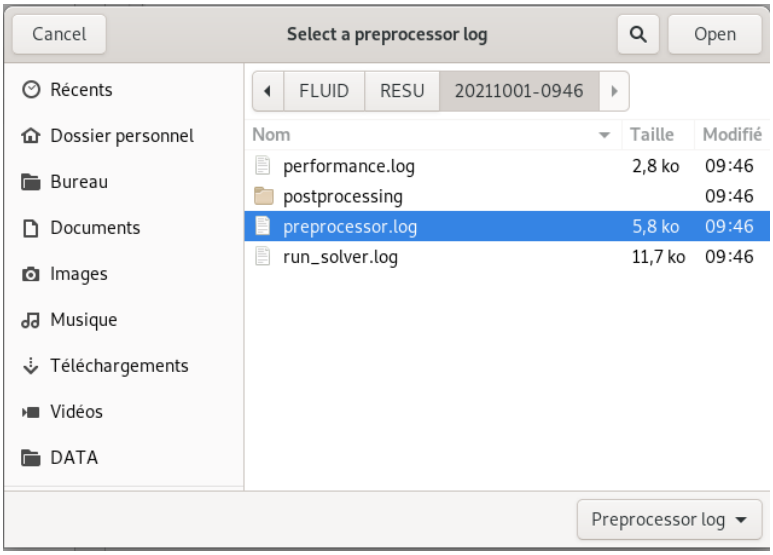

Figure III.18: Preprocessor.log fil loading window

Boundary zones Once the boundary regions automatically loaded , define the boundary conditions as follows

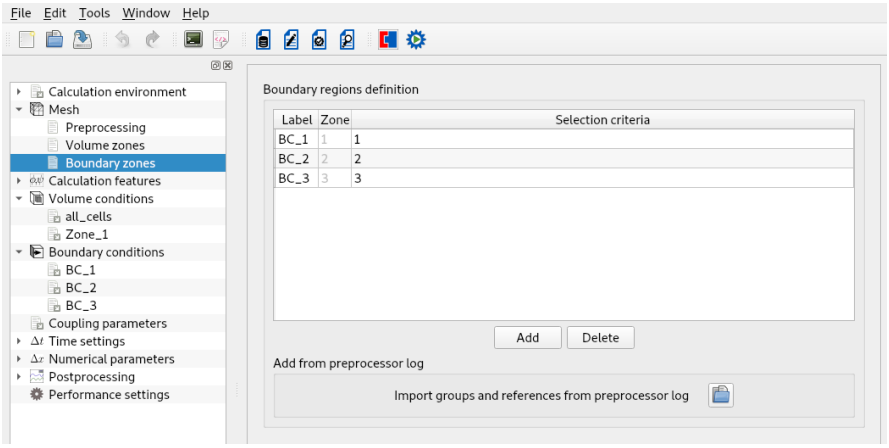

Figure III.19: Loaded Boundary zones

Rename selection criteria and delete the third zone.

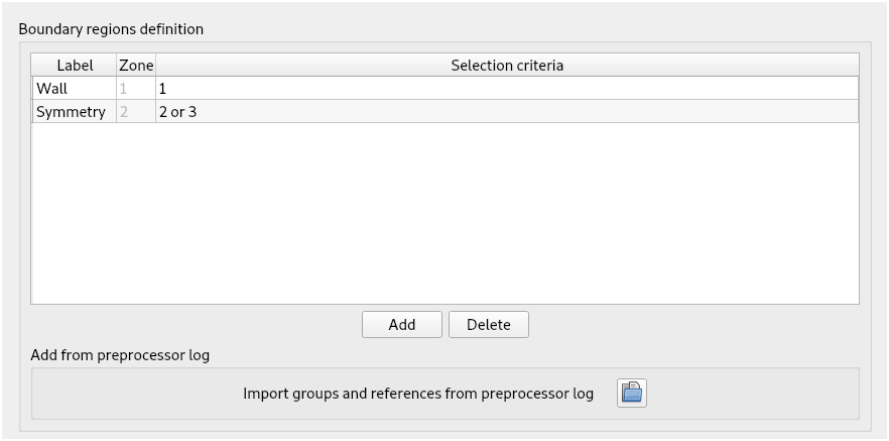

Figure III.20: Boundary zones updated

1.11 Calculation features Tab

• **Step 5**: Define Calculation features
Select Incompressible Standard Eulerian single phase Flow Models as follows

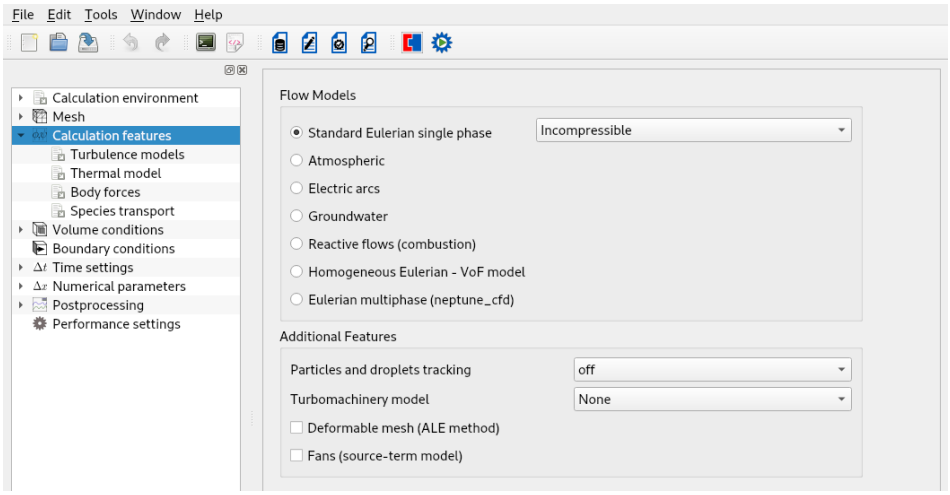

Figure III.21: Calculation features window

Turbulence models Select the $k-\varepsilon$ LP as turbulence model

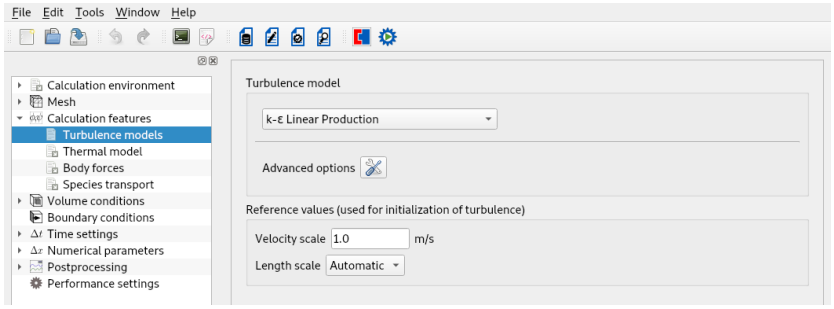

Figure III.22: Turbulence models

Thermal model Select Temperature (Celsius) under Thermal Scalar header

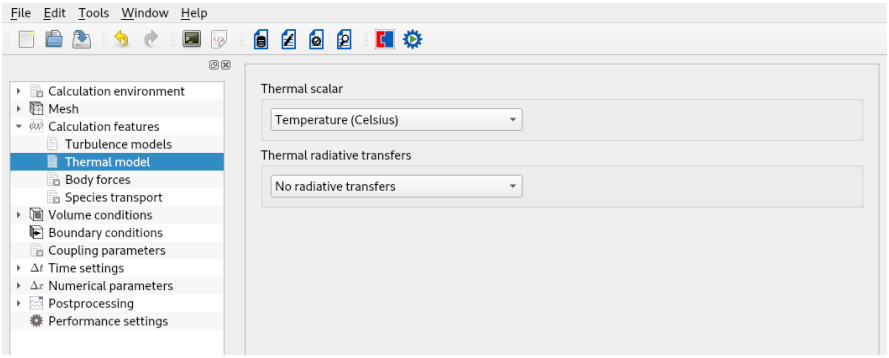

Figure III.23: Thermal Scalar

Body forces Add 1 G acceleration value following y axis as follows

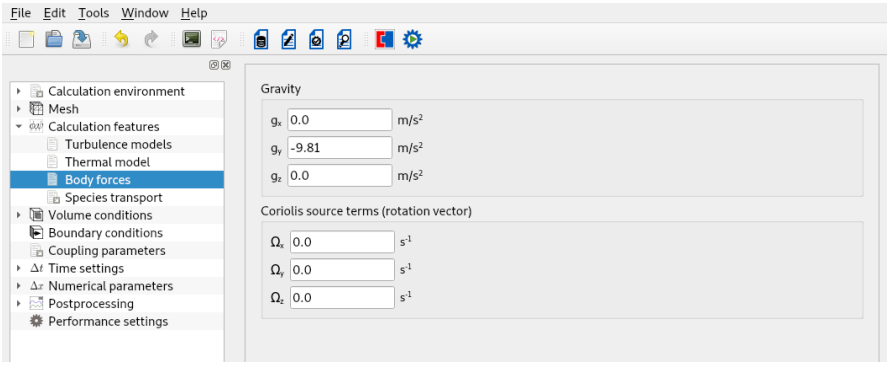

Figure III.24: Gravity settings

1.12 Volume conditions

- **Step 6**: Define Volume conditions

Tick Initialization and Physical properties

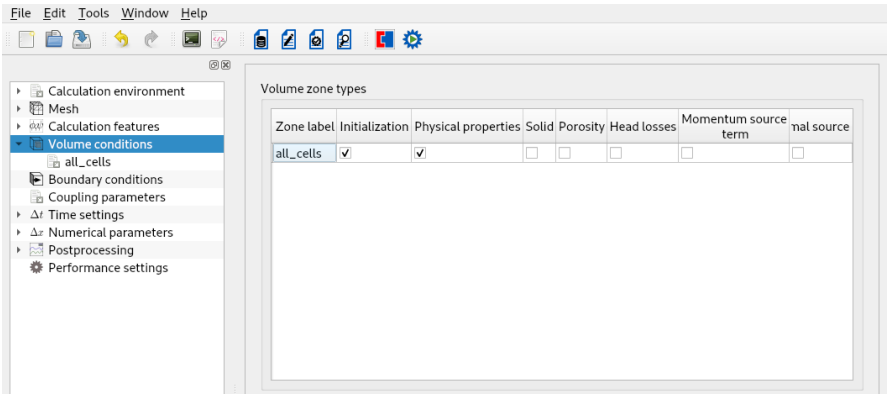

Figure III.25: Volume zone types

Initialization Initialize temperature value to 20°C

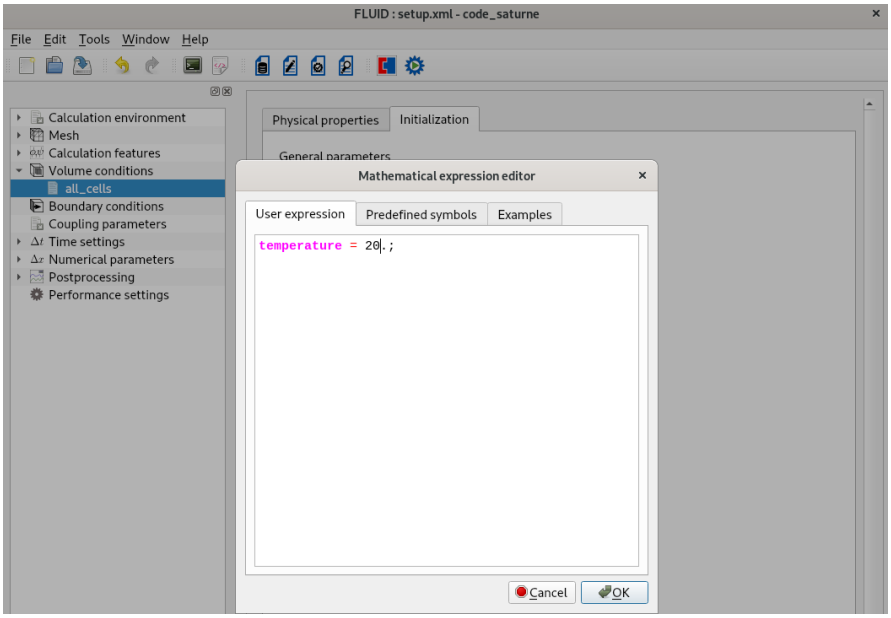

Figure III.26: Initialization of the temperature.

Physical properties Define all physical properties as follows

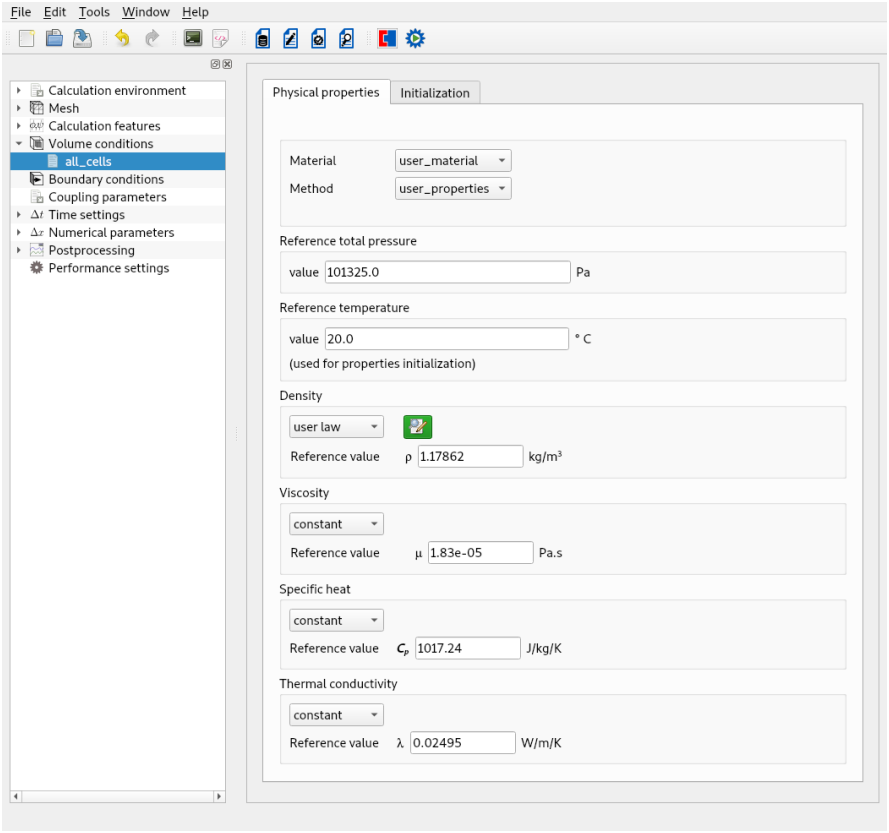

Figure III.27: Define the physical properties.

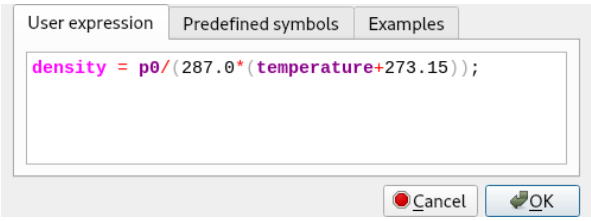

Figure III.28: Define the variable density with an ideal gas law.

1.13 Boundary conditions

• **Step 7**: Define Boundary conditions
First, check boundary zone types. Parameters should be set as follows : Define a thermal transfer

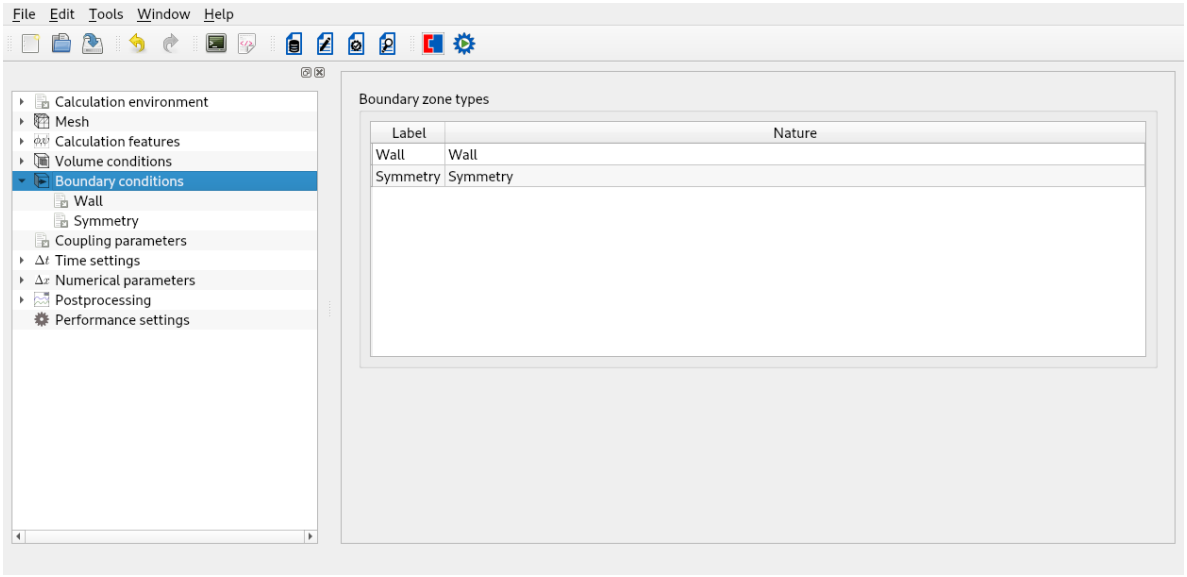

Figure III.29: Boundary conditions - Zone types

condition as wall boundary condition with an external wall temperature $T_{ext} = 30°C$and an exchange coefficient $q_{ext} = 10\ (W/m^2.K)$.

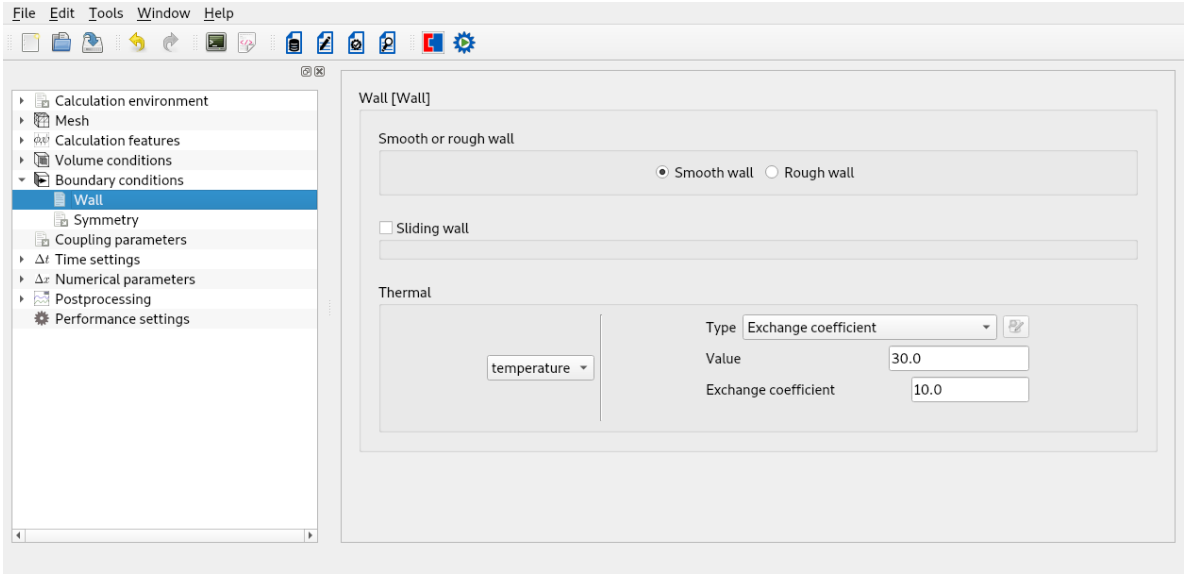

Figure III.30: Boundary conditions - Wall

1.14 Time settings Tab

• **Step 8**: Define the Time Settings

Figure III.31: Time settings windows

1.15 Postprocessing Tab

• **Step 9**: Define postprocessing
Define the writer and frequency output inside the code_saturne (GUI) as follows.

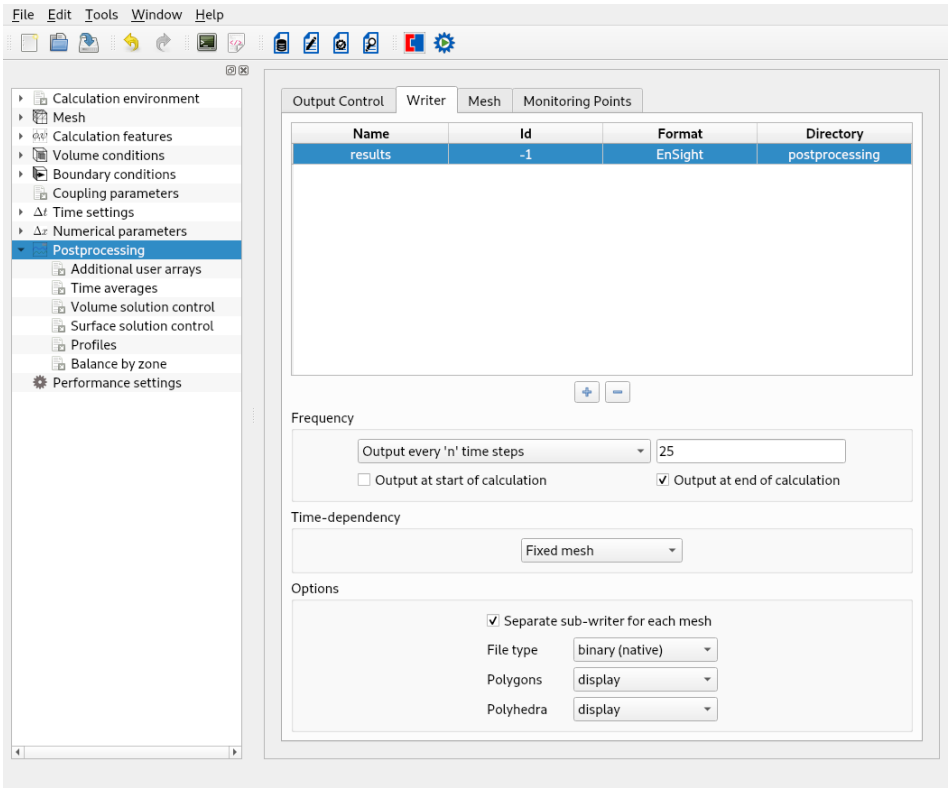

Figure III.32: Postprocessing - Writer Header

1.16 Run computation

• **Step 10**: Run the code_saturne computation alone. Launch the calculation by clickin on the 

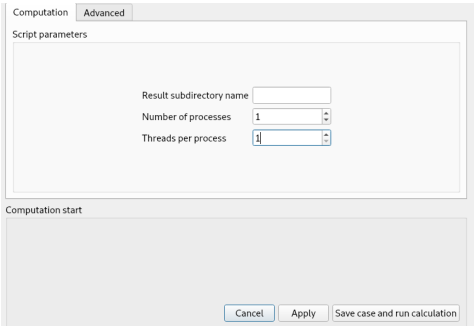

Figure III.33: Run computation

1.17 Postprocessing Analyses

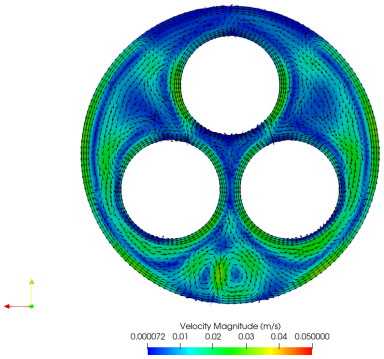

Figure III.34: Visualization of the 2D fluid velocity field

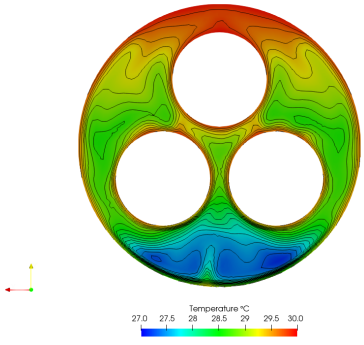

Figure III.35: Visualization of the 2D fluid temperature field

1.18 Preparing and launching code_saturne-SYRTHES coupled computation

In order to create a coupled compution between SYRTHES and code_saturne some settings must be applied.

The last modification to prepare the coupling computation are given below:
- **Step 1**: Activate conjugate heat transfer in SYRTHES GUI,
- **Step 2**: Activate conjugate heat transfer in code_saturne GUI,
- **Step 3**: Give identical number of iterations and reference time step for both codes,
- **Step 4**: Check the `run.cfg` python script and launch the calculation by executing the `runcase`.

You will, first, set SYRTHES parameters then code_saturne parameters.
- **Step 1**: Activate conjugate heat transfer in SYRTHES GUI

1.19 Coupling SYRTHES- Syrthes coupling

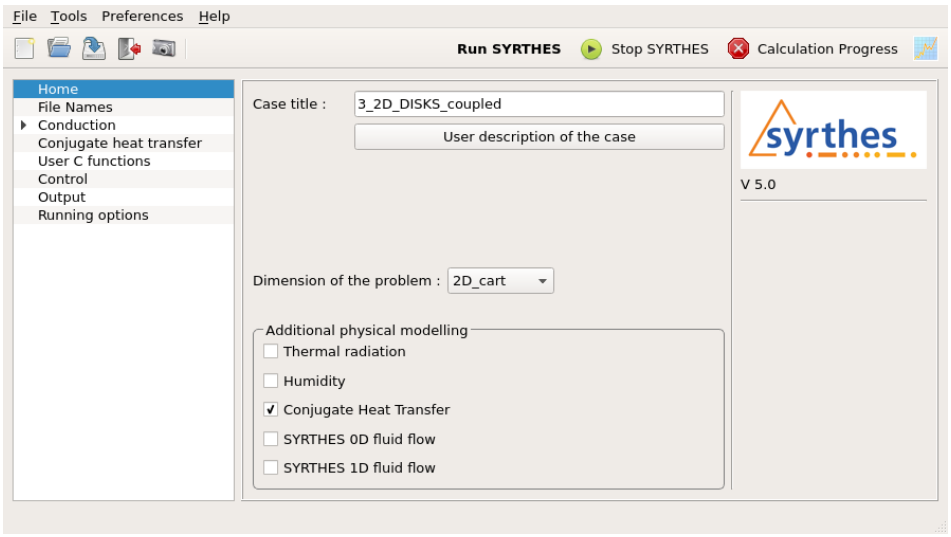

Figure III.36: SYRTHES Home Window - Conjugate Heat Transfer option ticked

Specify the references of the boundary zones for the coupling surfaces with code_saturne.

1.20 Coupling SYRTHES- Conjugate heat transfer Tab

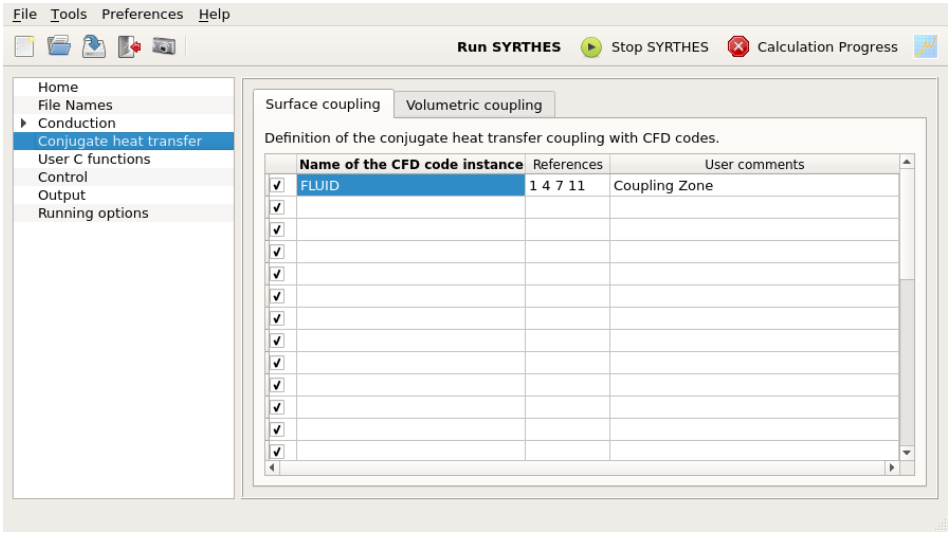

Figure III.37: Conjugate heat transfer tab - Surface coupling

1.21 Coupling SYRTHES- Control Tab

Change the number of iterations and reference time step for the solid domain.

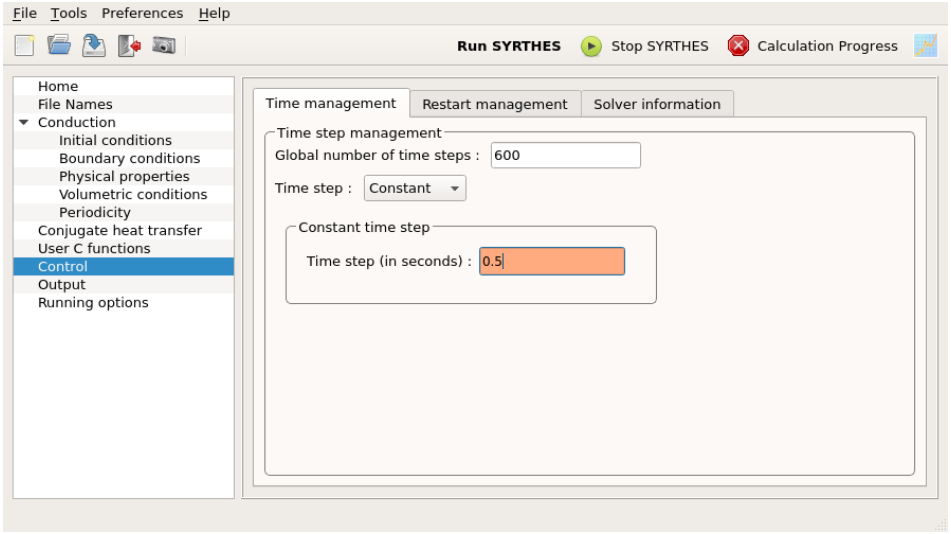

Figure III.38: Control Tab - Time Management

This step closes SYRTHES settings. Now you need to set code_saturne parameters.

• **Step 2**: Activate conjugate heat transfer in code_saturne GUI,

1.22 Coupling code_saturne- Boundary conditions Tab

First, you must activate **SYRTHES coupling** under **Boundary conditions** and fill SYRTHES instance field. The latest must be the name of the SYRTHES case previously created. In our turorial the case name is : **SOLID**

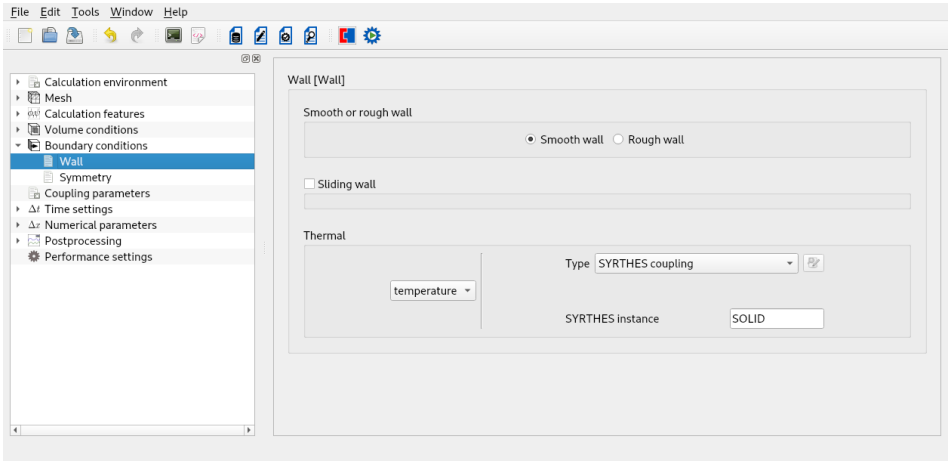

Figure III.39: Activate Thermal SYRTHES Coupling

1.23 Coupling code_saturne- Coupling parameters Tab

Set SYRTHES parameters as follows :

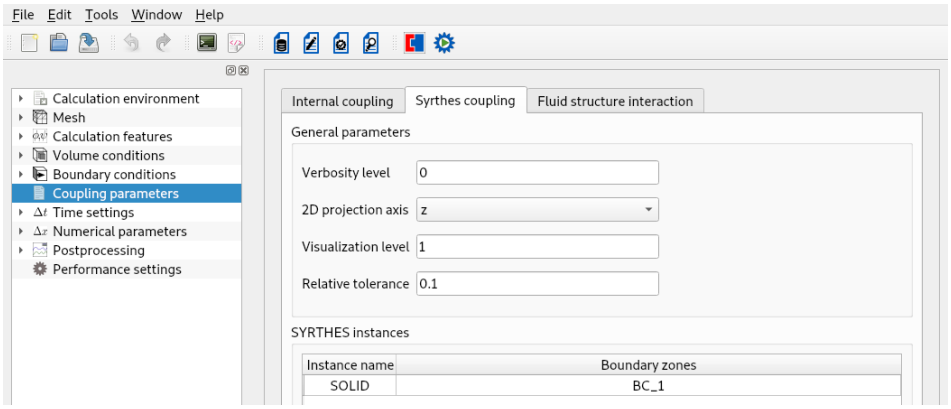

Figure III.40: code_saturne SYRTHES Coupling parameters

• **Step 3**: Give identical number of iterations and reference time step for both codes,

1.24 Coupling code_saturne- Time settings Tab

Change the number of iterations and reference time step for the fluid computation. It is important to set the same values as SYRTHES time settings.

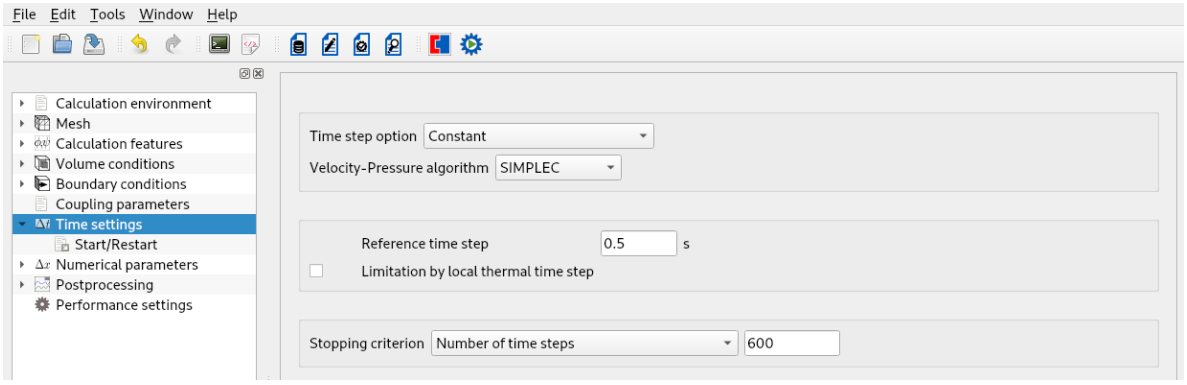

Figure III.41: Equalizing time settings parameters

1.25 Coupling code_saturne- Numerical Parameters Tab

Activate the **Improved pressure interpolation**.

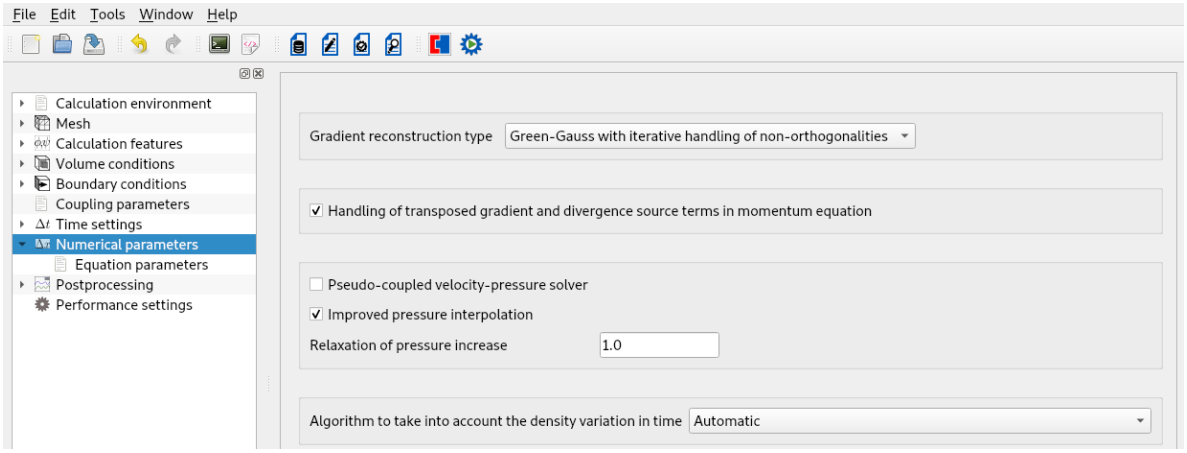

Figure III.42: Numerical parameters updated

• **Step 4**: Check the `run.cfg` python script and launch the calculation by executing the `runcase`.

1.26 Coupling Running computation

• **Substep 4.1 - Check/Edit run.cfg file settings**: After having enabled conjugate heat transfer on both sides (in the fluid and in the solid data sets), having set equal time step parameters on both sides and having checked/edited (if needed) the `run.cfg` script as follows:

```
>[setup]
>coupled_domains:  FLUID:SOLID

>n_procs:
>n_threads:

>[fluid]

>domain:  FLUID
>n_procs_weight:  1
>solver:  Code_Saturne

>[solid]

>domain:  SOLID
>n_procs_weight:  1
>opt :  -v ens
>param:  3_2D_DISKS_coupled.syd
>solver:  SYRTHES
```

you can process to the launch of computation.

Substep 4.2 - Run the case: Verify the **run.cfg** file is well filled. `names` and `param:` should be updated.

Finally, launch the `runcase` present in the study directory (named in our case 🗀`3_2D_DISKS`) and run the coupling computation via a terminal command as follows:

```
$ code_saturne run
```

• **Note:**: in the `run.cfg`, the number of processors can be specified for each code (as this example with 1 processors for code_saturne and 1 processors for SYRTHES). It can be either both codes in parallel, one in parallel and the other one in sequential, or both in sequential.

One can specify the ouput results format for SYRTHES with an option (`opt`) which takes the value `-v ens` for a 3D fields output with a EnSight format or `-v med` for a 3D fields output with a SALOME format).